

Demonstration of CFD to support the coupled analysis of a reactor pressure vessel subjected to pressurized thermal shock

Nuclear Science and Engineering Division

About Argonne National Laboratory

Argonne is a U.S. Department of Energy laboratory managed by UChicago Argonne, LLC under contract DE-AC02-06CH11357. The Laboratory's main facility is outside Chicago, at 9700 South Cass Avenue, Argonne, Illinois 60439. For information about Argonne and its pioneering science and technology programs, see www.anl.gov.

DOCUMENT AVAILABILITY

Online Access: U.S. Department of Energy (DOE) reports produced after 1991 and a growing number of pre-1991 documents are available free at OSTI.GOV (<http://www.osti.gov/>), a service of the US Dept. of Energy's Office of Scientific and Technical Information.

Reports not in digital format may be purchased by the public from the National Technical Information Service (NTIS):

U.S. Department of Commerce
National Technical Information Service
5301 Shawnee Rd
Alexandria, VA 22312
www.ntis.gov
Phone: (800) 553-NTIS (6847) or (703) 605-6000
Fax: (703) 605-6900
Email: **orders@ntis.gov**

Reports not in digital format are available to DOE and DOE contractors from the Office of Scientific and Technical Information (OSTI):

U.S. Department of Energy
Office of Scientific and Technical Information
P.O. Box 62
Oak Ridge, TN 37831-0062
www.osti.gov
Phone: (865) 576-8401
Fax: (865) 576-5728
Email: **reports@osti.gov**

Disclaimer

This report was prepared as an account of work sponsored by an agency of the United States Government. Neither the United States Government nor any agency thereof, nor UChicago Argonne, LLC, nor any of their employees or officers, makes any warranty, express or implied, or assumes any legal liability or responsibility for the accuracy, completeness, or usefulness of any information, apparatus, product, or process disclosed, or represents that its use would not infringe privately owned rights. Reference herein to any specific commercial product, process, or service by trade name, trademark, manufacturer, or otherwise, does not necessarily constitute or imply its endorsement, recommendation, or favoring by the United States Government or any agency thereof. The views and opinions of document authors expressed herein do not necessarily state or reflect those of the United States Government or any agency thereof, Argonne National Laboratory, or UChicago Argonne, LLC.

Demonstration of CFD to support the coupled analysis of a reactor pressure vessel subjected to pressurized thermal shock

prepared by

Yiqi Yu, Dillon Shaver, and Elia Merzari

Nuclear Science and Engineering Division, Argonne National Laboratory

May 26, 2021

Abstract

The structural components that comprise nuclear reactors and their supporting structures are subjected to harsh operating environments that can challenge their integrity, especially after exposure for extended duration or under accident condition. As one of the most significant components of a reactor, the Reactor Pressure Vessel (RPV) is exposed to an aggressive environment during the operation time (e.g. more than 40 years). Aging degradation mechanisms (e.g. thermo-fatigue) could grow initial defects up to a critical size, increasing the susceptibility to failure in the RPV. The conventional methods are mostly based on simple crack and structure geometries. Very limited studies consider the real conditions of the RPV subjected to a thermal shock due to a Loss of Coolant Accident (LOCA). During a LOCA event, the most severe conditions take place when the emergency core cooling (ECC) water is injected inside the cold legs filled initially with hotter water and/or steam. The rapid cooling of the down-comer and the internal RPV surface followed probably by re-pressurization of the RPV causes large temperature gradients and variation of pressure which induces thermal-mechanical stresses. In order to develop the model for integrity assessment of a reactor pressure vessel (RPV) subjected to pressurized thermal shock (PTS), a multi-physics simulation, which includes the thermo-hydraulic, thermo-mechanical and fracture mechanics analyses is necessary. The prediction of the temperature field is achieved by using computational fluid dynamics (CFD) simulation. In this report, a demonstration CFD standalone simulation is performed to support coupled analysis for Reactor Pressure Vessel (RPV) subjected to Pressurized Thermal Shock (PTS). The study use a simplified computational domain to represents a real RPV. The purpose of the study is to demonstrate the transient temperature response of RPV to ECC injection. The CFD model is built in a robust and efficient way for further coupled calculation. The next steps of this work, including the coupled thermal and tensor mechanics capabilities using Cardinal are expected to be complete by the end of FY21 for the demo problem. After this, into FY22, the capability will be demonstrated for a realistic RPV.

Contents

Abstract	i
1 Introduction	1
2 Model Description	2
2.1 Geometry and Mesh	2
2.2 Initial and Boundary Conditions	3
3 Results and Discussion	5
4 Conclusions and Continuing Work	9
Acknowledgments	12
References	14

List of Figures

2.1	Geometry of the computational domain	3
2.2	Mesh structure	4
2.3	Boundary conditions of the CFD model	5
3.1	Velocity distribution on different cross sections	6
3.2	Temperature evolution on the center cross section	7
3.2	Temperature evolution on the center cross section (cont.)	8
3.3	Temperature evolution on the inner wall of the RPV	10
3.4	Locations of the monitor points on the inner wall of the RPV	11
3.5	Temperature development of the monitor points on the inner wall of the RPV	11

List of Tables

2.1	Geometric parameters of the CFD model	2
2.2	Fluid properties, inlet, and initial conditions for the CFD model	4

1 Introduction

The Reactor Pressure Vessel (RPV) is a very important component in a nuclear power plant, the structural integrity of which should be maintained throughout the whole plant life. The RPV may suffer high thermal stresses due to extreme temperature gradients caused by rapid cooling under the conditions of a Loss of Coolant Accident (LOCA) transient [1]. The critical operating conditions, including high temperature and internal pressure associated with light water reactors (LWRs), also brings a great threat to the structural integrity of the RPV. The combination of high internal pressure and thermal stresses can cause possible crack propagation through the vessel wall if defects exist in the RPV. Aside from that, the material properties are subject to degradation during reactor operation by neutron irradiation, fatigue, thermal ageing and other mechanisms, reducing the resistance of the RPV against brittle fracture. Understanding the thermal stresses is key to mitigating the risk associated with these failure mechanisms. Some CFD simulations [2, 3, 4, 5] have been applied to study the three-dimensional coolant mixing thermal hydraulic behavior in an RPV. These CFD calculated results were mostly within the uncertainty bands of experiments. More recently, a benchmark study was used to validate Nek5000 for buoyancy-driven mixing in a cold leg-downcomer region [6, 7]. In these studies, turbulent mixing of two miscible fluids of differing densities was simulated. The simulations showed good agreement with available experimental data and demonstrated the need for a high-fidelity capability to accurately capture mixing phenomena. In sum, the reliability of CFD methods has been verified widely around the world. A coupled multi-physics approach with high fidelity simulations is regarded as one of the most promising directions to realize the improvement of computational accuracy for nuclear power system research.

The Pressurized Thermal Shock (PTS) phenomenon requires a strong coupling between thermal hydraulic and structure analysis. The study of RPV under PTS loading has attracted the attention of international experts dedicated to the integrity assessment, especially because of the thermal hydraulic aspects. There is significant current interest in the development of advanced reactors or research reactors, nearly all of which operate at significantly higher temperatures than LWRs, and many of which include design features, such as molten salt coolants, that impose additional challenges on structural materials. Many international projects were organized to bring together experts to perform detailed analyses. These include previous efforts by the OECD/NEA [8, 9] to provide benchmark cases for thermal fatigue. These studies focus on transient thermal-mechanical loading conditions expected in potential loss-of-coolant accidents. The earlier report defines an assessment in a hypothetical RPV, while the latter focuses on a simple pipe undergoing cyclic cooling due to a water injection.

The main outcome of a TH analysis is the thermal and pressure loads affecting the RPV in case of a relevant transient. Extreme thermal gradients in the structural components can take place during PTS. Therefore, the fluid temperature must be reliably assessed to predict the loads upon the RPV. Conventionally, one dimensional system codes such as RELAP and TRACE are widely employed for thermo-hydraulic calculation. However, these transient simulations are reduced to simplified one-dimensional or axisymmetric cases, disregarding the real temperature distribution. Unlike these codes, high fidelity methods allow the detailed geometry to be taken into account and to predict multidimensional features of PTS in the RPV. In particular, CFD is proven to be

able to represent the PTS temperature evolution in detail in the coolant during the transient [10]. Thus, the application of CFD has the ability to represent three-dimensional features of the flow that directly affect the structural response. When coupled with a capability to predict thermal stresses in the RPV, this can be a very useful tool in the prediction of thermal fatigue phenomena. In this report, NekRS will be used for a CFD simulation to predict detailed three-dimensional flow pattern that cannot be predicted properly with one-dimensional codes. This simulation will later be leveraged using the tensor mechanics module of the MOOSE framework via the Cardinal code [11] to demonstrate a capability for thermal striping.

2 Model Description

2.1 Geometry and Mesh

As shown in Figure 2.1a, a simplified geometry is adopted for computational domain to mimic a portion of an RPV. The cold leg, a wall of the neutron shield and the RPV are represented in a simplified way for further detailed modeling of a realistic RPV (Figure 2.1b). The neutron shield, located in the active area of the core, which provides shielding for the complete vessel, represents an obstacle to the flow in the downcomer. It has a significant influence on the mixing characteristics of the cold water falling down in the downcomer and thermal stresses in the RPV. As water is injected into the cold legs, the hot legs are not considered to be particularly relevant for the PTS investigation. Thus, they are not fully modeled and instead the outlet boundary condition is applied.

The dimension of the geometry is listed in Table 2.1. The parameters of the simplified model are chosen carefully to make sure they are as close to the actuality as possible. The diameter of the cold leg is 1 m while the length is 5 m. The total height of the domain 8 m. The center of the nozzle on the RPV wall is 2.5 m from the top of the domain and is 5.5 m from the bottom of the domain. The width of the plate is 3 m, which is close to $\frac{1}{4}$ (90°) of the arc length of the whole RPV for a PWR. The plate thickness is 0.25 m, which is close to the wall thickness of RPV. The thickness of the flow channel thickness is 0.35 m, which is close to the distance between RPV inner wall and the wall of neutron shield.

Table 2.1: Geometric parameters of the CFD model

<i>Parameters</i>	<i>value [m]</i>
Cold leg diameter	1
Cold leg length	5
Plate width	3
Top part height	2
Bottom part height	5
Flow channel thickness	0.35
RPV thickness	0.25

The entire computational domain consists of both a fluid region and a solid region. The CFD calculation only contains the fluid region. The solid region will be taken into account in MOOSE for

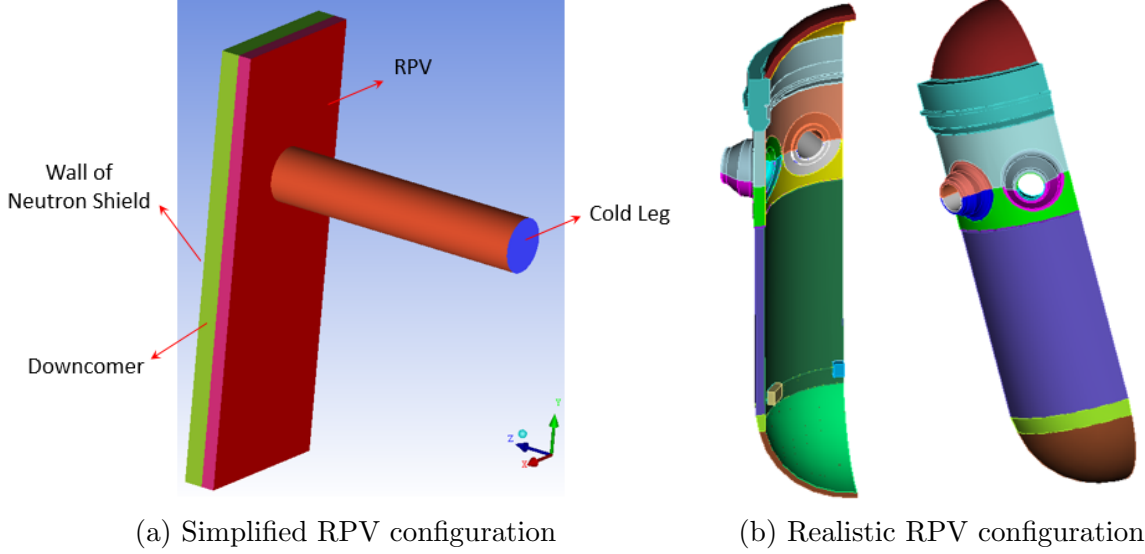


Figure 2.1: Geometry of the computational domain

a further coupled calculation. A computational grid of hexahedral cells is used in NekRS to obtain the flow field. The mesh structure is shown in Figure 2.2. The appropriate mesh refinements in zones of crucial importance for the PTS analysis are applied. The total number of elements in fluid region is 63,548 and the total number of nodes is 73,768. The relatively fewer number of elements in the domain for this simplified geometry can make the calculation efficient and quick, which benefits the testing for the model. Since the mesh for the realistic RPV geometry can contain many more elements, the strategy is to use this demonstration case as a starting point for the coupled capability.

2.2 Initial and Boundary Conditions

The most relevant initial and boundary conditions are summarized in Table 2.2. The whole domain is filled up with stationary water at saturation temperature. The pressure is assumed to be uniform. The injection temperature is assumed constant. The flow is assume to be laminar to save the computational cost. The parameters in the solver of NekRS is nondimensionalized with the following equation:

$$V^* = V/V_0 \quad (1)$$

$$T^* = T/T_0 \quad (2)$$

Where V^* and T^* are non-dimensional velocity and temperature.

The inlet of the cold leg is set as a Dirichlet boundary condition with constant velocity. The top and bottom of the flow channel are set as pressure outlet boundaries. The side boundaries are a set of periodic boundary conditions. The boundary conditions are shown in Figure 2.3. The interfaces between solid and fluid region are set as wall boundary conditions in CFD simulation. As the cold water (10°C) is injected into the hot domain (280°C), the temperature in the whole computational

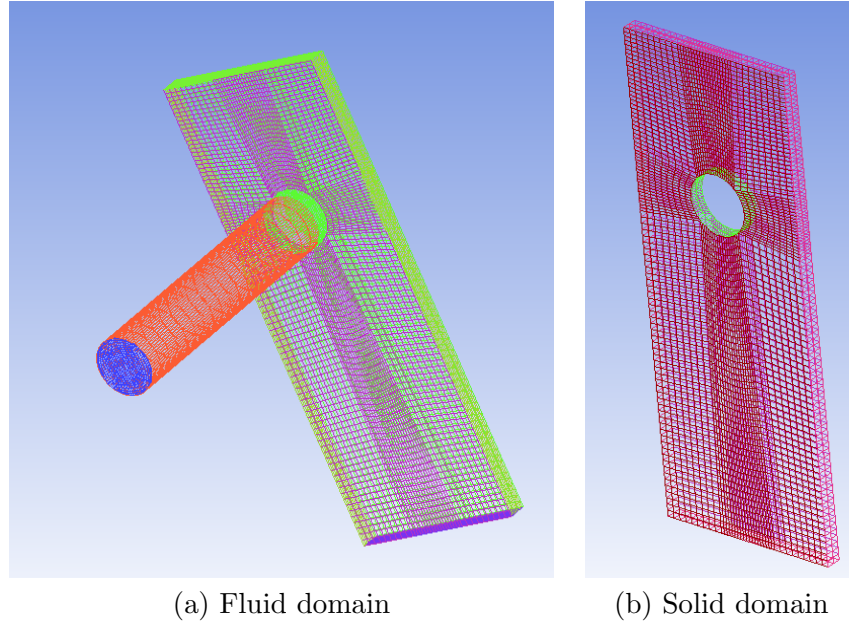


Figure 2.2: Mesh structure

Table 2.2: Fluid properties, inlet, and initial conditions for the CFD model

<i>Parameters</i>	<i>value</i>
pressure	6MPa
viscosity	0.2Pa-s
density	1000kg/m ³
specific heat	4200J/kg-K
thermal conductivity	0.6W/m-K
Reynolds number	500
Prandtl number	10
Peclet number	5000
inlet velocity (V_0)	0.102m/s
inlet Temperature (T_0)	10°C
inlet mass flow rate	80kg/s
initial velocity	0m/s
initial temperature	280°C
ΔT	270°C

domain will begin to decrease. After a long time of ejection, the temperature of the whole domain will finally reach the temperature of inlet injection.

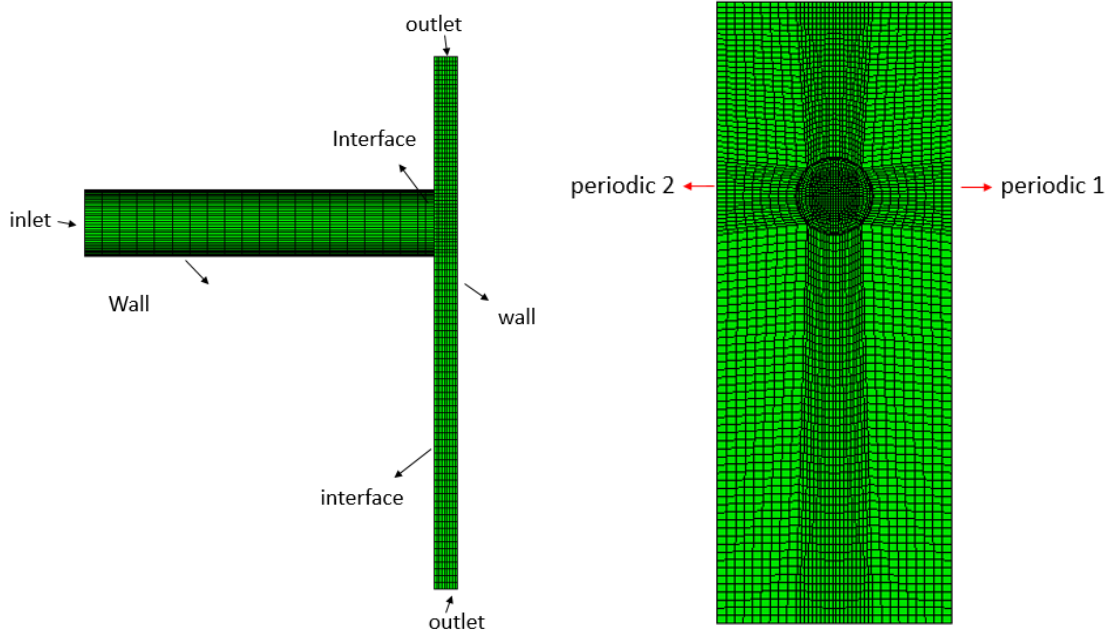


Figure 2.3: Boundary conditions of the CFD model

3 Results and Discussion

Since the initial velocity in the whole domain is 0, the cold water ejected into the domain will develop along the flow path. When the flow is full developed, the velocity distribution in the domain will reach a quasi-steady state. Figure 3.1 shows the velocity distribution on the center cross section. The velocity profile develops into a steady laminar-flow profile in the cold leg until the flow impinges on the flat plate. The flow is divided into two streams and a zero velocity zone is formed near the impinging region. The velocity near the junction between the cold leg and RPV wall is small.

Figure 3.2 shows the temperature evolution on the center cross section. The temperature in the cold leg begin to drop as soon as the cold water is ejected. After 4 seconds, the cold water reaches the junction. The temperature in the flow channel will also drop quickly. The heat transfer mainly relies on convection. After about 20 seconds, the cold water will reach the bottom of the downcomer. By this time, most of the domain is cooled down to the inlet temperature. However, some locations on the RPV inner wall still remain hot as the velocity near the RPV wall is small. From 20 to 40 seconds, the temperature of the RPV inner wall will decrease slowly due to conduction.

The temperature on the RPV inner wall is critical for stress analysis. Figure 3.3 shows the temperature evolution on the inner wall of RPV. The temperature of RPV inner wall remains at the initial temperature until the cold water reach the junction (4s). The temperature near the

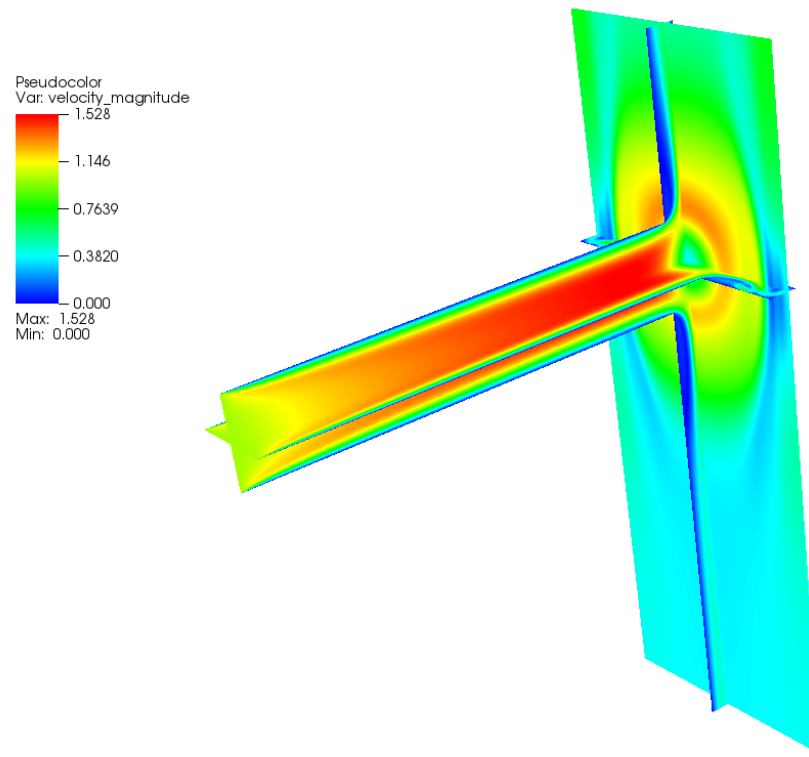


Figure 3.1: Velocity distribution on different cross sections

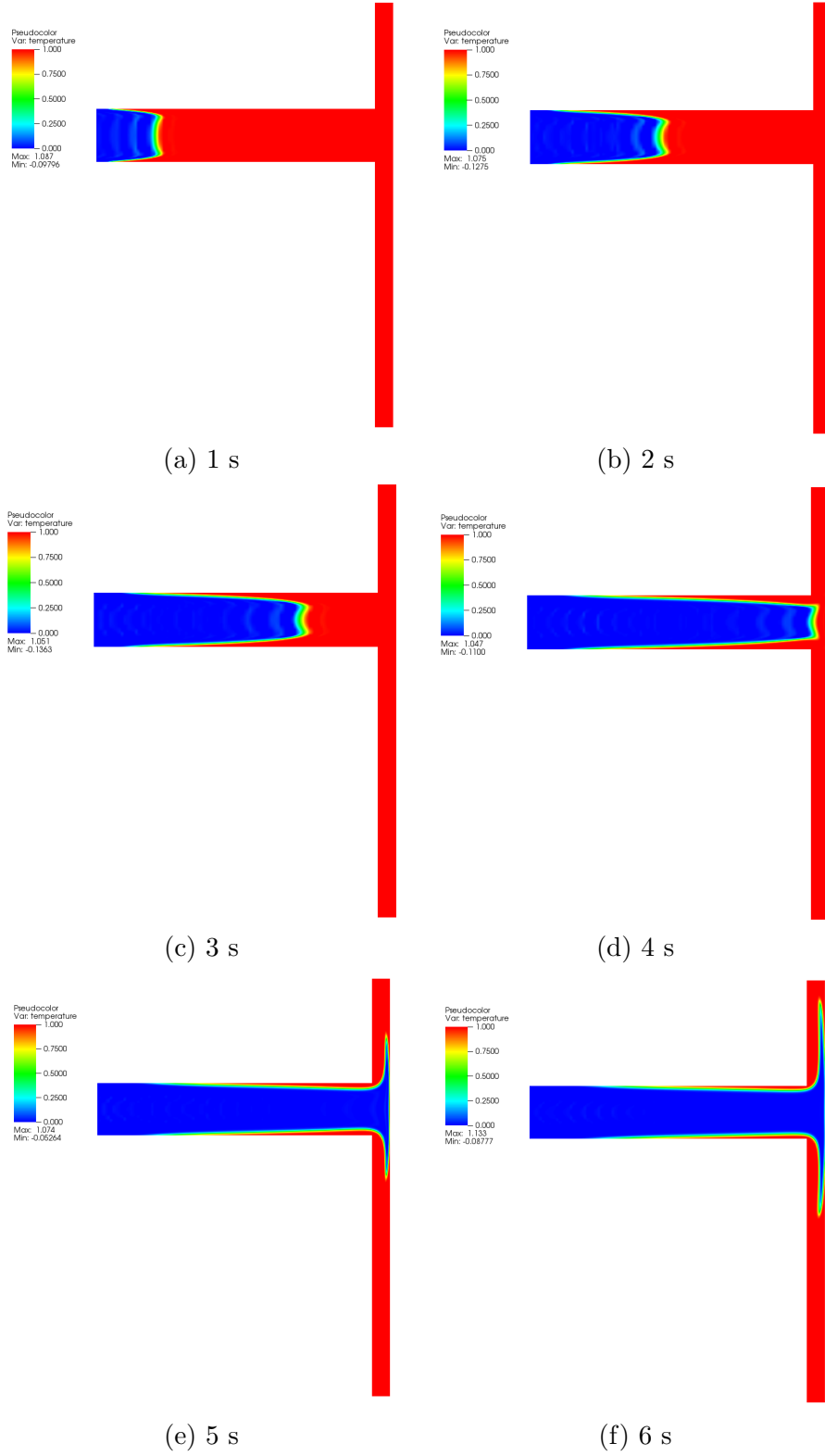
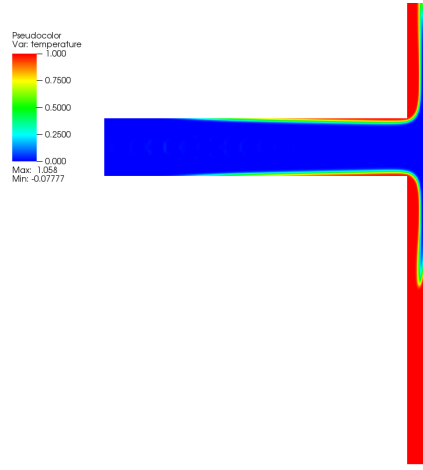
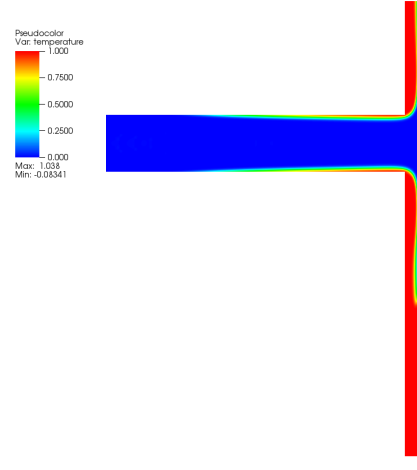


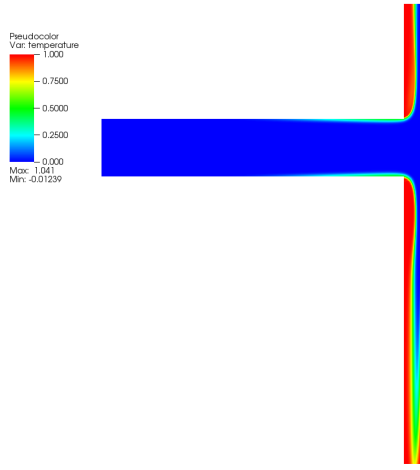
Figure 3.2: Temperature evolution on the center cross section



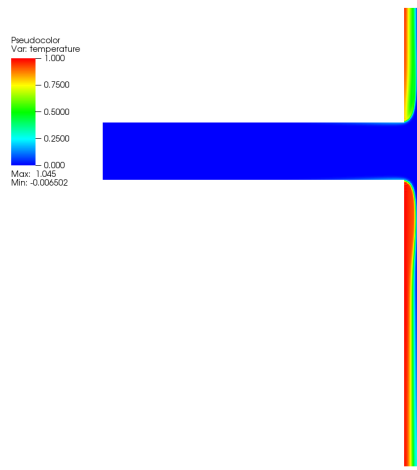
(g) 7 s



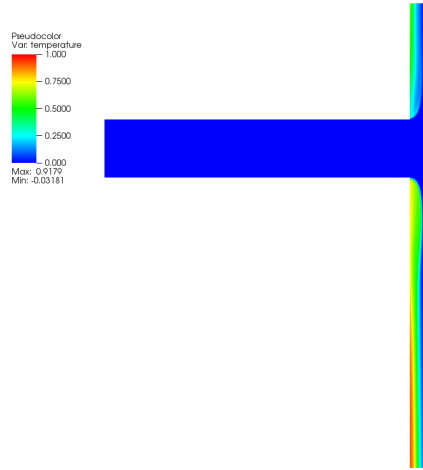
(h) 8 s



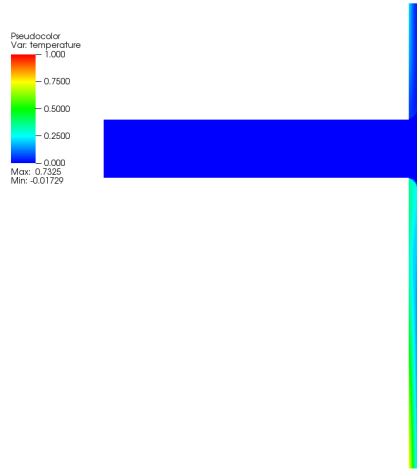
(i) 15 s



(j) 20 s



(k) 30 s



(l) 40 s

Figure 3.2: Temperature evolution on the center cross section (cont.)

impinging area drops to the inlet temperature quickly. Then the area about 1 diameter on either side of the impinging area begins to be cooled down first. Two symmetric cooling plumes are formed. These two plumes then expand along the sides of the downcomer. It is demonstrated that the PTS temperature evolution during the transient is well represented using the three dimensional CFD calculation and the cooling plume is determined in detail.

The aim of the CFD study is to provide the thermal load to the stress analyses. In order to better understand the temperature evolution, six monitor points are set on the RPV wall. The detailed location of these monitor points are shown in Figure 3.4.

Figure 3.5 shows the temperature development during the 40s transient at the monitor points. Since the flow is laminar, no temperature fluctuations are observed. The temperature of the left and right monitor points decrease to the inlet temperature about 20s after the injection. It takes a longer time for the center point to be cooled down. The higher monitor points always have a delay on temperature drop. The RPV wall shows symmetric temperature development as the temperature history of the left and right monitor points are identical. The patterns are generally similar for the left, central and right monitor point, while the exact timings of the temperature decrease or increase differ. The different temperature gradients as functions of time will lead to different stress intensities in the RPV.

4 Conclusions and Continuing Work

In this report, a demonstration CFD standalone simulation was performed to support coupled analysis for a Reactor Pressure Vessel (RPV) subjected to Pressurized Thermal Shock (PTS). The study uses a simplified computational domain to represents a real RPV. The transient temperature response of the RPV to ECC injection is predicted by CFD simulation, which is critical for predicting the features of stress intensities in the RPV. The temperature evolution on the inner wall of the RPV shows a symmetric behavior. Several monitor points are setup to observe the temperature history. The patterns are generally similar for the left, central and right monitor points, while the exact timings of temperature decrease or increase differ. The temperature differences, which are hard to obtain by other conventional methods, will cause different thermal loads on the wall. Besides that, the mesh strategy and numerical scheme used in this demonstrational case will also be a good reference for CFD simulations on the realistic RPV geometry.

The next steps for this work will be to integrate the conjugate heat transfer and tensor mechanics capabilities of MOOSE via Cardinal. This will provide solutions for the temperature distribution and thermal stresses in the solid domain. The coupled thermal capability with Cardinal is expected to be demonstrated by the end of July 2022, with a complete demo incorporating the tensor mechanics expected by the end of FY21. Into FY22, effort will then focus on demonstration for a realistic RPV, similar to that shown in Figure 2.1b.

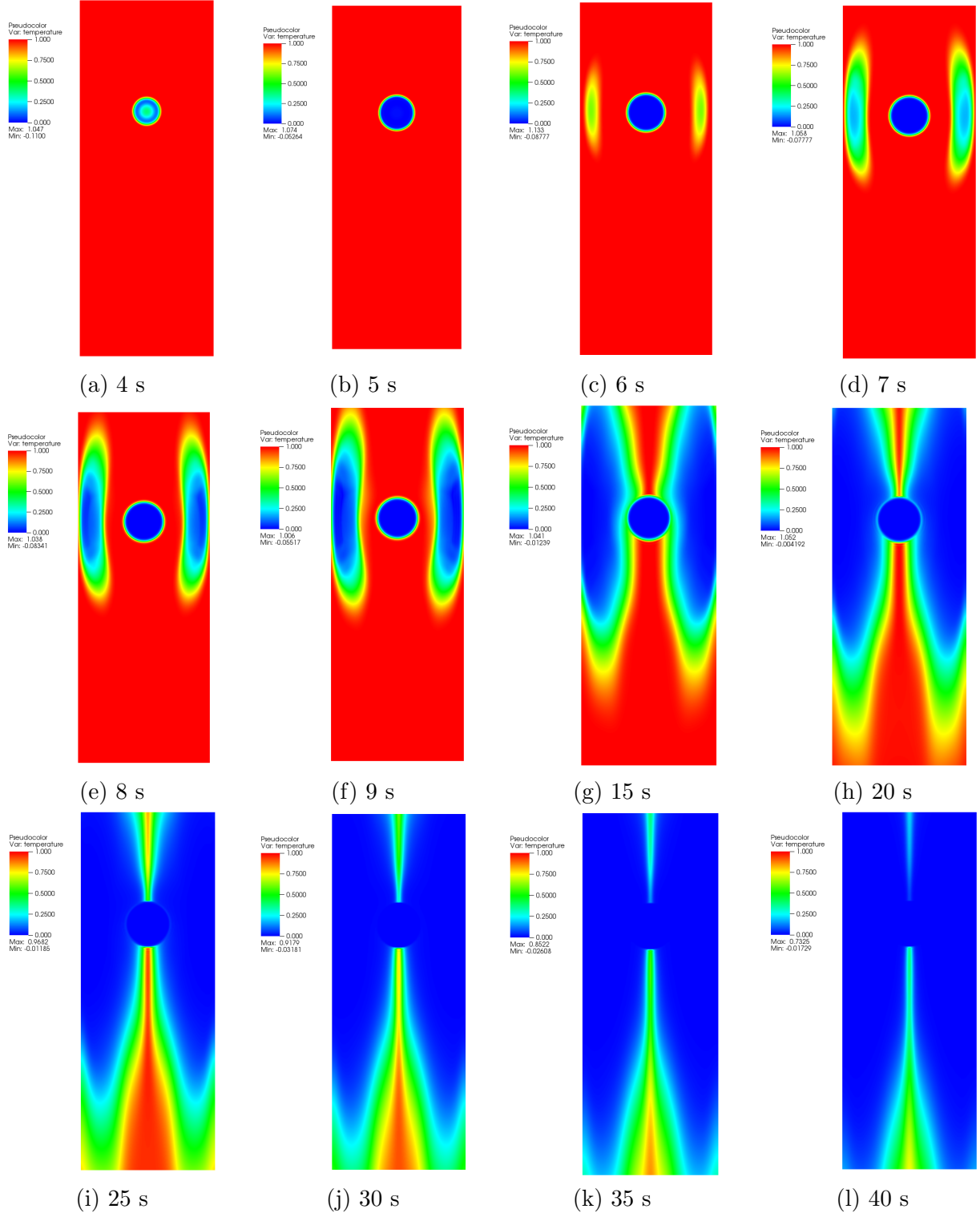


Figure 3.3: Temperature evolution on the inner wall of the RPV

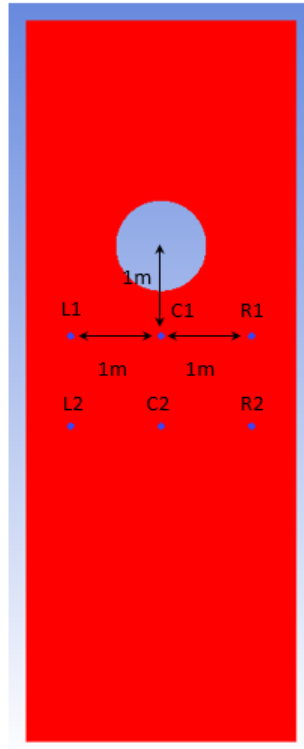


Figure 3.4: Locations of the monitor points on the inner wall of the RPV

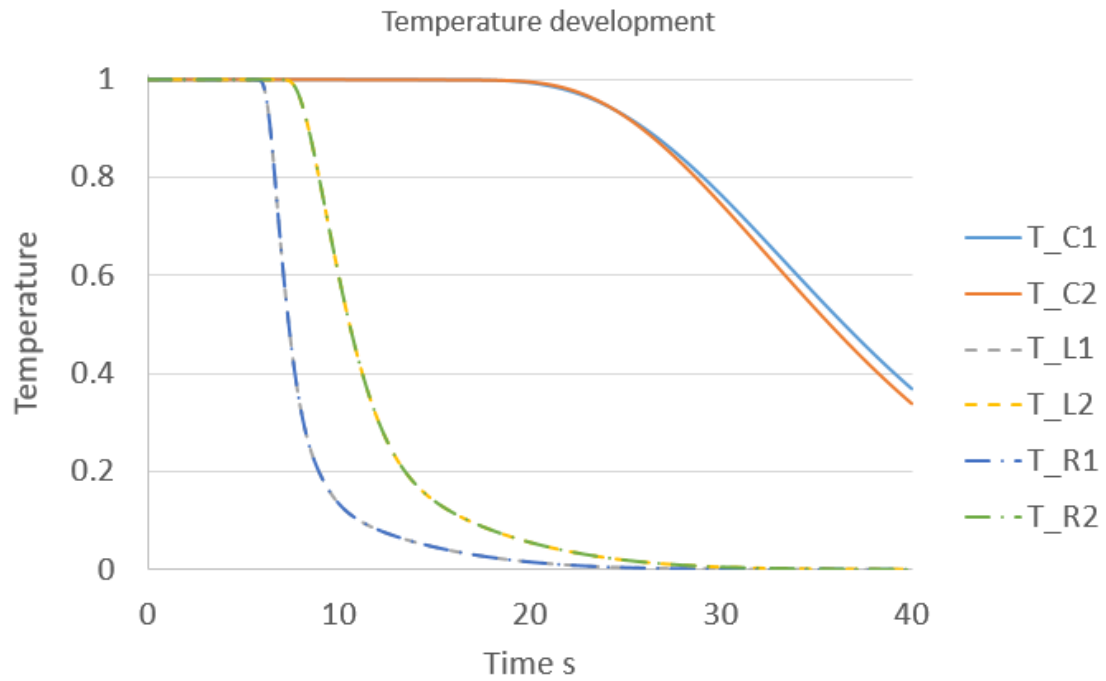


Figure 3.5: Temperature development of the monitor points on the inner wall of the RPV

Acknowledgments

Argonne National Laboratory's work was supported by the U.S. Department of Energy, Office of Nuclear Energy, Nuclear Energy Advanced Modeling and Simulation (NEAMS) program, under contract DE-AC02-06CH11357.

References

- [1] D. Lucas and D. Bestion, “On the simulation of two-phase flow pressurized thermal shock (PTS),” in *The 12th International Topical Meeting on Nuclear Reactor Thermal Hydraulics (NURETH-12)*, (Pittsburgh, PA), 2007.
- [2] J. N. Reyes, J. T. Groome, A. Y. Lafi, D. Wachs, and C. Ellis, “PTS thermal hydraulic testing in the OSU APEX facility,” *Int’l J. Pressure Vessels and Piping*, vol. 78, pp. 185–196, 2001.
- [3] T. Toppila, “CFD simulation of Fortrum PTS experiment,” *Nucl. Eng. and Design*, vol. 238, pp. 514–5221, 2001.
- [4] S. M. Willemsen and E. M. J. Komen, “Assessment of RANS CFD modeling for pressurized thermal shock analysis,” in *The 11th International Topical Meeting on Nuclear Reactor Thermal Hydraulics (NURETH-11)*, (Avignon, France), 2005.
- [5] U. Rhode, T. Höhne, S. Kliem, B. Hemström, M. Scheuerer, T. Toppila, A. Aszodi, I. Borose, I. Farkas, P. Mühlbauer, L. Vyskocil, J. Klepac, J. Remis, and T. Dury, “Fluid mixing and flow distribution in a primary circuit of a nuclear pressurized water reactor – validation of CFD codes,” *Nucl. Eng. and Design*, vol. 237, pp. 1639–1655, 2007.
- [6] J. K. Lai, E. Merzari, and Y. A. Hassan, “Sensitivity analysis in a buoyancy-driven closed system with high resolution CFD using Boussinesq approximation and variable density models,” *Int’l J. Heat and Fluid Flow*, vol. 75, pp. 1–13, 2019.
- [7] J. K. Lai, E. Merzari, Y. A. Hassan, P. Fischer, and O. Marin, “Verification of validation of large eddy simulation with Nek5000 for cold leg mixing benchmark,” *Nucl. Eng. and Design*, vol. 358, 2020.
- [8] “Comparison report of RPV pressurized thermal shock international comparative assessment study (PTS ICAS),” Tech. Rep. NEA/CSNI/R(99)3, Paris, France, Nov. 1999.
- [9] “FAT3D- an OECD/NEA benchmark on thermal fatigue in fluid mixing areas,” Tech. Rep. NEA/CSNI/R(2005)2, Paris, France, Aug. 2005.
- [10] M. Sharabi, V. F. González-Albuixech, N. Lafferty, B. Niceno, and M. Niffenegger, “Computational fluid dynamics study of pressurized thermal shock phenomena in the reactor pressure vessel,” *Nucl. Eng. and Design*, vol. 299, pp. 136–145, 2016.

- [11] E. Merzari, H. Yuan, M. Min, D. Shaver, R. Rahaman, P. Shriwise, P. Romano, A. Talamo, Y.-H. Lan, D. Gaston, R. Martineau, P. Fischer, and Y. Hassan, “Cardinal: A lower length-scale multiphysics simulator for pebble-bed reactors,” *Nuclear Technology*, pp. 1–23, 2021.



Nuclear Science and Engineering Division

Argonne National Laboratory
9700 South Cass Avenue, Bldg. 208
Argonne, IL 60439

www.anl.gov



Argonne National Laboratory is a U.S. Department of Energy
laboratory managed by UChicago Argonne, LLC